International Journal of Mechanical and Production Engineering Research and Development (IJMPERD)
ISSN(P): 2249–6890; ISSN(E): 2249–8001
Vol. 9, Issue 5, Oct 2019, 1077–1088
© TJPRC Pvt. Ltd.

TRANS STELLAR
•Journal Publications •Research Consultancy

Original Article

SIMULATION IN HEAT TRANSFER ENHANCEMENT OF COMPACT HEAT EXCHANGER USING ALUMINUM OXIDE (Al_2O_3) AND MAGNESIUM OXIDE (MgO) USING CFD

S. S. ATHIB MALICK[1] & C. SADHASIVAM[2]

[1]*Student, Department of Mechanical Engineering, Saveetha Institute of Medical and Technical Sciences, Chennai, Tamil Nadu, India*

[2]*Assistant Professor, Department of Mechanical Engineering, Saveetha Institute of Medical and Technical Sciences, Chennai, Tamil Nadu, India*

ABSTRACT

In this present work, the cross flow heat exchanger (CFHE) is utilized and computational fluid dynamics (CFD) is used to analyze the performance of evaporator to enhance the thermal efficiency, and to improve the distribution of fluid flow. CFHE has a capacity of providing 1.5 m^3/min of water at 55°C for a Combined Heat & around 200°C for the nano fluids. The CFD principles utilized in this effort are effective and suitable techniques that use finite elements to solve the heat exchanger processes. The required finite element computations are done by Ansys software tool and the obtained outputs are shown in graphical representation. This analysis is accomplished for this heat exchanger under the required conditions of inlet flow to the evaporator, and finally the comparison and conclusion are presented.

KEYWORDS: Cross Flow Heat Exchanger; Computational Fluid Dynamics & Flow Characteristics

Received: Jul 11, 2019; **Accepted:** Aug 03, 2019; **Published:** Sep 28, 2019; **Paper Id.:** IJMPERDOCT201996

Nomenclature

A_F	Surface area of fins, mm^2
D_F	Diameter of fin, mm
A_t	Surface area of tube, mm^2
R	Reynolds number
V_m	Maximum velocity, mm/s
D	Inside diameter of the tube
μ	Dynamic viscosity
ρ	Density, kg/m^3
N_u	Nusselt number
h	Convective heat transfer coefficient, W/m^2K
k	Thermal conductivity, W/mk
Q	Heat transfer, W
η	Efficiency of fin

1. INTRODUCTION

A heat exchanger device is employed for transferring the thermal energy in between two or additional liquids. Heat exchangers area units, utilized in each heating and cooling practices [1–4]. The fluids are also divided by a firm partition to stop combining or they will be in surface contact [5–7]. They are widely utilized in the applications such as air conditioning & refrigeration, oil & gas refineries and waste treatment. The typical

example of a device is initiated in an inside combustion engine, through the coils of radiator and air flows past the coils, that lower the temperature of the fluid, and heats the received air. Additional instance is that the sink, which may be a passive device that transfers the heat produced by a robot to a liquid medium.

Shell and tube heat exchangers formed with a number of tubes, in which, the fluid or liquid that has to be cooled or heated. The fluid flows over the surface of tubes that has to be heated or cooled, and therefore it can be either heated or absorbing the heat required [8]. Generally, a series of tubes is termed as bundle of tubes, and it can be formed into different varieties such as lengthwise fin and plain. It is usually employed for the aggressive areas with the temperature higher than 260°C and the pressure higher than 30 bars. So for that, it is strong because of their shapes. Various design of heat exchangers should be supposed about once forthcoming with tubes [9]. The tubes are in various forms such as U tube, straight and bent forms.

Most of the investigators are previously reported to examine the flow features and transferring of heat in heat exchangers applications. Generally, due to the application of centrifugal forces, the heat transfer performance in helical heat exchanger system is enhanced. Because of the tube curvature along with a circulatory motion, a secondary flow field is generated which leads to moving the liquid elements towards the inner region of the tube. This flow field increases rates of heat transfer by decreasing the temperature slope across the tube cross-section. Therefore, there is an occurrence of extra convection heat transfer, which is the right angle to the axis of the main flow but it does not occur in the straight tube heat exchanger.

Bharuka et al., [10] reported the heat transfer features in double pipe heat exchanger experiments. It was reported that while increasing of Nusselt number, the overall heat transfer coefficient enhanced. But, it has increased as a purpose of the rate of mass flow. Kumar et al., [11] reported the performance of pressure drop and heat transfer features in tube-in-tube heat exchanger experiments. CFD software was utilized to find the flow and heat transfer in the experiments. The friction factor and Nusselt number were reported in the literatures that are compared with the output of experimentation. The simulation of CFD was in good validation with the experimental outputs.

Ghorbani et al., [12] varied the following parameters such as Reynolds number, different coil pitch, and different ratios of tube-coil diameter to study the convection heat transfer rate in a heat exchanger. This research work was conducted based on both laminar and turbulent flow through the inner coil. The influences of tube diameters and coil pitch on the heat transfer coefficient of the exchanger were investigated. It was reported that the surface area of coil was the important and significant parameter on the output parameter, and influence of diameter of tube was minimal due to that, it was almost negligible.

Kharat et al., [13] studied the heat transfer amount for concentric helical coils heat exchanger. The already existed correlation was found to be a large discrepancy in the results, due to the high gap between the coils in the heat exchanger. This study was used with the CFD software package for predicting the heat transfer coefficient. The mathematical modeling was also developed to analyze the output of the data from the simulation software. The high correlation exists in between the numerical technique and experimental outputs within an error of 3–4 %.

Jayakumar et al., [14] investigated a helical coil heat exchanger by CFD software. This study was concentrated on the analysis, based on the restrictions of constant heat flux and temperature. It was found that the constant temperature boundary conditions for heat exchanger did not provide the required modeling. According to the experimental outputs, they generated a good relationship to find the coefficients of heat transfer in the coil. From this research work, Nusselt number values were obtained at a high range in outer coil and lower in the inner side of

the coil. Also, the parameters such as pitch circle diameters and pipe diameters are identified as influencing parameters on the heat transfer coefficients in the heat exchanger.

Amar et al., [15] studied the experimental work of double pipe heat exchanger, in which, the heat transfer amounts were determined for both parallel and counter flow with the limits of constant temperature and heat flux. The temperature was measured based on the variation of flow rates in the inside tube and in the annulus. Using Wilson plots, the overall heat transfer coefficient in the inside tube and annulus were recorded. Also, Nusselt numbers were found for both annulus and inner tube.

2. GOVERNING EQUATIONS

The heat transfer analysis from the cross-flow heat exchanger involves solving 2nd order differential equations. It is often a subject of investigators comprising heat transfer coefficient as a function of geometrical dimensions of fins. Generally, the research of the heat transfer in the extended surface compromises the heat movement within fin, through the conduction process and the heat transfer from the fin to the surroundings through the convection process [16–18]. For the analysis of heat transfer, the following geometrical properties of heat transfer are required on each side of the heat exchanger.

The following equations are needed to calculate the surface area of fins and tubes.

Surface area of fins is expressed by equation (1)

$$A_F = \pi D_F \delta + \left\{ {}^{1}/_{2}\, \pi (D_F^2 - D^2) \right\} \quad (1)$$

Surface area of tube is expressed by equation (2)

$$A_t = \pi D s \quad (2)$$

Reynolds number is defined by equation (3)

$$R = {}^{\rho V_m D}/_{\mu} \quad (3)$$

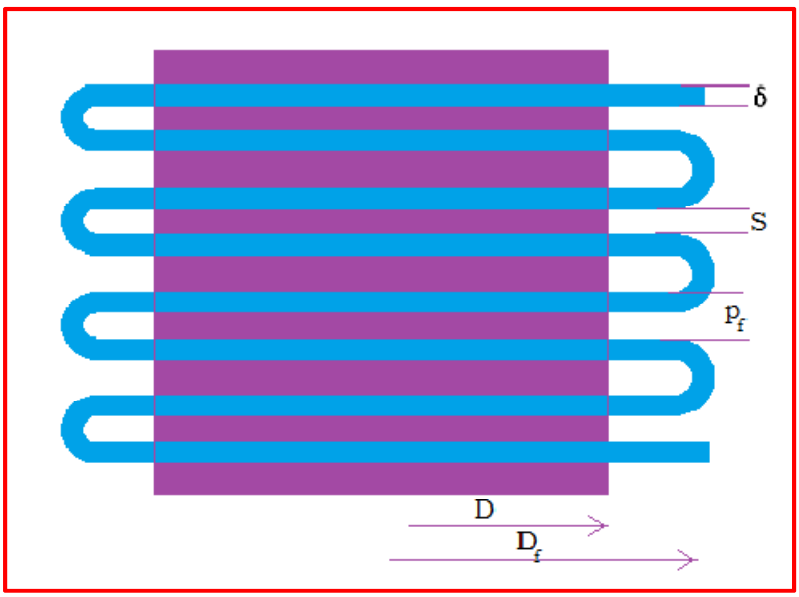


Figure 1: Geometry of Fin-Tube Heat Exchanger [18].

Maximum velocity of fluid is defined by equation (4)

$$V_m = {}^{m}/_{A\rho} \tag{4}$$

Nusselt number can be calculated by equation (5)

$$N_u = {}^{hD}/_{k} \tag{5}$$

Effective heat transfer coefficient can be expressed by equation (6) for the fluid flowing out and at perpendicular to the axis of tubes.

$$h = 5.3\frac{V_{in}^{0.59}}{D^{0.39}}\left[\frac{X}{X-D}\right]0.59 \tag{6}$$

Net heat transfer can be determined with the consideration of efficiency of fin (η)

$$Q = h\Delta T(\eta A_F + A_t) \tag{7}$$

where

$$\Delta T = \frac{[T-T_{out}]-[T-T_{in}]}{Y}; Y = ln\frac{T-T_{out}}{T-T_{in}}; \tag{8}$$

After transformation equation (8), the following equation can be obtained

$$\Delta T = (T_{in} - T)\frac{1-exp\left\{-\frac{h(\eta A_F+A_t)}{mC_F}\right\}}{\frac{h}{mC_F}(\eta A_F+A_t)} \tag{9}$$

The efficiency of fin can be calculated from equations

$$\eta = \frac{tanh\sqrt{x}}{\sqrt{x}}; x = \frac{2h}{\delta k}\psi \tag{10}$$

$$\psi = {}^{D}/_{2}\left[\frac{D_F}{D} - 1\right]\left[1 + 0.4ln\frac{D_F}{D}\right] \tag{11}$$

3. METHODOLOGY

This work was started with designing a cross flow heat exchanger, using CFD, capable of providing 1.5 m^3/min of water at 55°C for a Combined Heat & around 200°C for the nano fluids. To solve this task, a theory based approach was used to identify values of importance like the relationship between the (T_{in}) inlet, and (T_{out}) outlet temperatures, the overall, U, and Cp, heat transfer quantities, and the Q heat transfer rate for the fluids used; these were later inputted to Ansys Workbench. In ANSYS software, precaution to ensure the results were as accurate as possible and performed mesh refinement studies at key points, to ensure mesh independence.

Analysis was started from a simple 2D model and the CFD results easily with expected results from theoretical calculations, and progressively moved to the final 3D model. To achieve the requirements, variable factors affecting heat exchange are considered (e.g. pipe spacing, angle of attack) independently, and later used the best performing designs of each factor, in devising a closed 3D solution to the problem. In conclusion, a successful design of heat exchanger capable of heating water at room temperature to 55°C (328.15K) using CFD simulation developed. However, although the design was analysed and adjusted for any discrepancies between the CFD approximation and the real world part, the design brief lacked a number of real world specification, which lead to a more accurate design. In addition to it, we have added internal fins to disturb the incoming the fluid, so that we can reduce the temperature of the fluid. The methodology followed in this work is shown in figure 2.

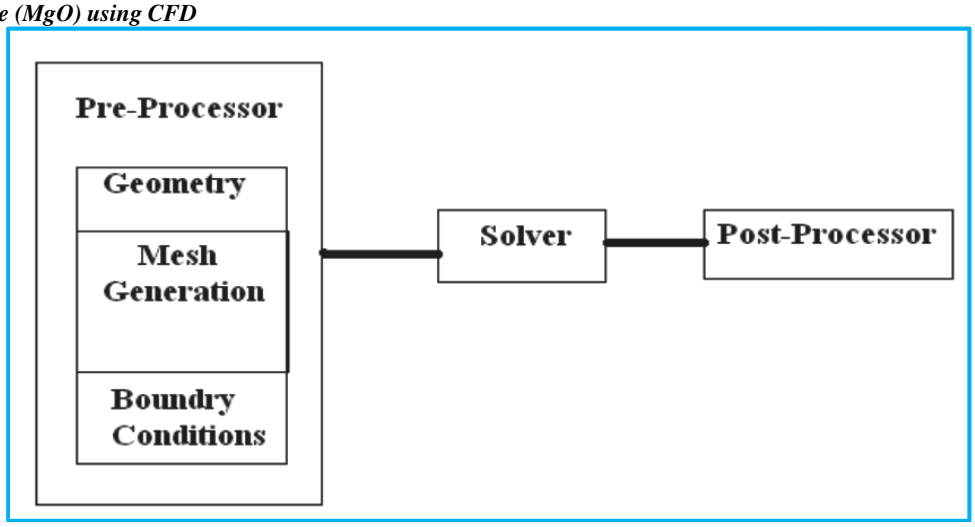


Figure 2: Methodology of CFD Simulation.

3.1. Pre-Processing

Step1: Construction of geometry. This work has three geometries such as cold fluid section, pipe section and cold fluid section.

The geometrical models are modelled using CATIA V5 software tool. Then, this model is imported in ANSYS workbench software for CFD analysis. Fluid flow module (FLUENT) is chosen from this workbench.

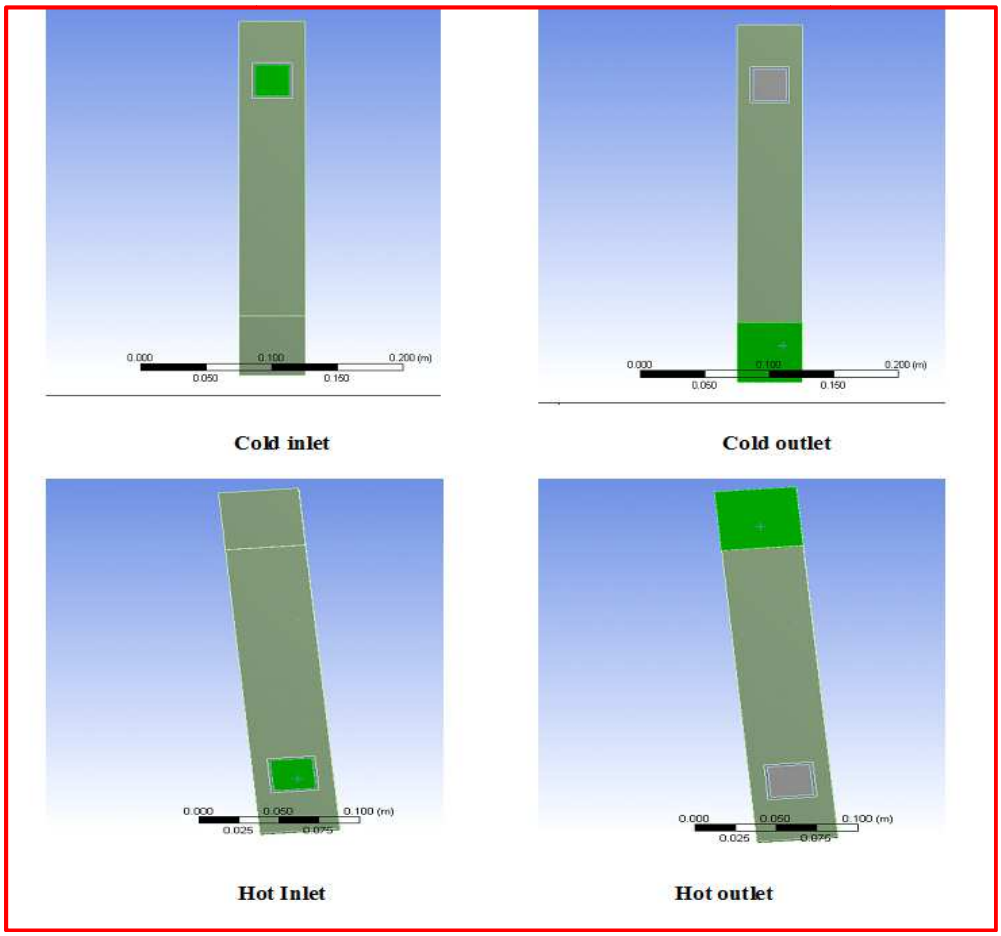


Figure 3: Cross Flow Heat Exchanger with Cold Flow Inlet & Outlet and Hot Flow Inlet & Outlet.

3.1.1. Merging

The three different models were modeled by using CATIA software. Then, all the models are merged into a single part using a control option (figure 3). At the end of this process, the one part with 3 bodies is available to investigate the CFD analysis. The three bodies or components were identified as follows: 1. Inner fluid 2. Tube 3, Outer shell.

3.1.2. Meshing

The most important step in the stage of pre-processing is mesh generation, since it used to fit the boundary conditions. Generally, mesh generation is needed in many engineering applications, which are appropriate for the problem solving with three dimensional Navier-Stokes equations. In the present work, the tetrahedron and hexahedral elements were selected for three dimensional meshing which have quadrilateral and triangular shapes at the limits. At last, a fine meshing is formed with edge sizing, and it has regions of high temperature gradients. The net amount of elements is 120000 and number of interior faces is 48000.

3.1.3. Creating Named Sections

According to the present research work, the various sections are named as cold_IN, cold_OUT, hot_IN and hot_OUT. The assignment was saved, updated and then meshed window are shut down at the end of the operation. Then, fluent launcher in ANSYS was opened by clicking on the setup box.

3.2. Solution

3.2.1. Problem setup

The meshing of the model was tested to improve the interpolation at the boundaries. Time was set as steady state condition. Select the analysis type as a pressure type and the velocity formulation is set as absolute formulation.

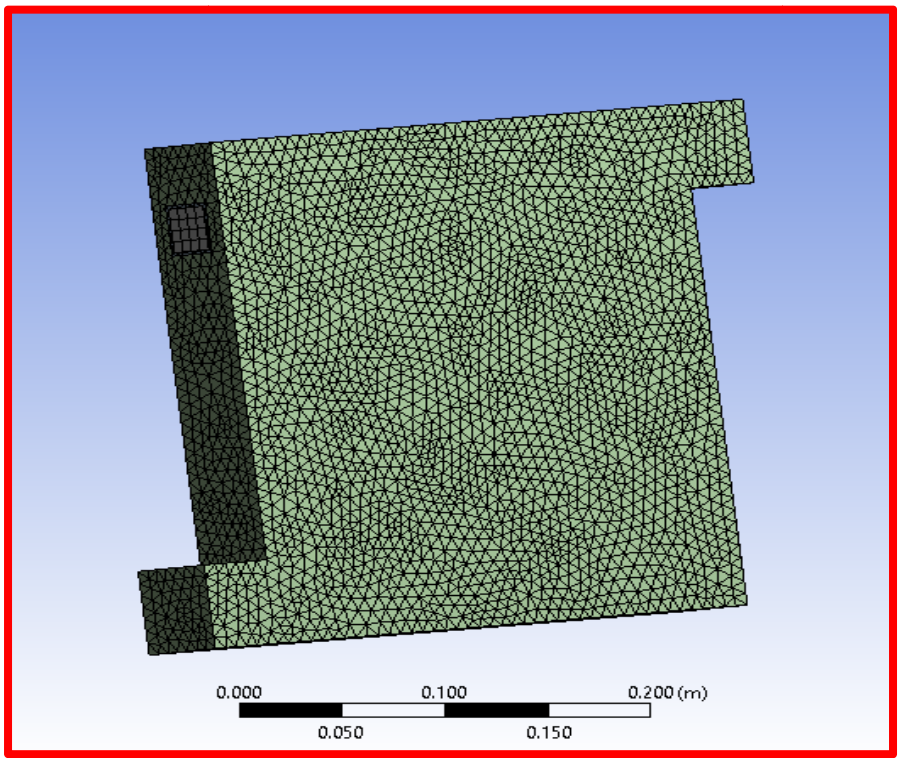


Figure 4: Meshed Design Structure.

3.2.2. Materials

Table 1 Geometrical Details of Heat Exchanger

Sl. No.	Particulars	Value
1	Number of baffles	4
2	Number of internal tubes	2
3	Tube length, mm	1500
4	Tube inner dia, mm	30
5	Tube outer dia, mm	35
6	Number of hot fluid inlet	2
7	Number of cold fluid inlet	1
8	Dimension of cold and hot fluid, mm	150 * 800 and 400*150
9	Overall dimension, mm	1800*200*865
10	Longitudinal pitch, mm	45
11	Transverse pitch, mm	65
12	Dimension of baffles, mm	10*150*750
13	Number of separation plate, mm	4
14	Dimension of separation plate, mm	10*150*750
15	Material of tube, separation plate and baffle	Mild steel

Fluid medium for this analysis as water-liquid and Al_2O_3 and Cu was selected as solid elected from the ANSYS fluent database. In order to simplify the problem domain, the motor has not been included in the CFD model. For CFD analysis, the geometrical details of CFHE considered are given in table 1.

3.2.3. Assumptions

The following assumptions are considered during this CFD analysis.

- The loss of heat from the upper side of CFHE by natural convection. Rate of heat transfer coefficient was assumed at the upper side 2500 W/m^2K.
- Radiation heat from heat exchanger surface is neglected.

3.2.4. Boundary conditions

The various limits were applied for different zones of a heat exchanger (table 2). Because it is a type of tube-in-tube heat exchanger system, in this contains two inlets and two outlets. Velocity was assigned as inlets and pressure was assigned as outlets. The velocity of 2.5 m/s was maintained as constant for inlet velocity of the cold fluid, and from 1 to 2 m/s velocity was varied for the hot fluid. The atmospheric pressure was set as outlet pressure for the experiments. The inlet hot fluid temperature was kept at 650°C and inlet cold fluid temperature was kept at 230°C. The convective heat transfer parameter in between the outer surface and atmosphere was fixed at 2500 W/m^2K, and also surrounding temperature was set as 270 °C. The remaining conditions were defined accordingly.

Table 2 Boundary Conditions Considered during Analysis

Sl. No.	Boundary Conditions and its Units	Value
1	Cold fluid temperature, °C	230
2	Hot fluid temperature, °C	650
3	Surrounding temperature, °C	270
4	Convection heat transfer, W/m^2K	2500
5	Velocity, m/s	2.5

3.2.5. Solution methods

The simple scheme is utilized for all calculations. The least square cell based approximation was selected as gradients. The pressure and momentum are set as linear and second order upwind, respectively. Under relaxation factor of 0.75 was selected as high order term relaxation. The turbulent kinetic energy and dissipation rate are chosen as second order upwind.

3.2.6. Convergence conditions

The residuals ad moment coefficients are monitored. The convergence conditions for the three velocity components and continuity were set as 10^{-5}, for energy was set as 10^{-8} and for turbulent kinetic energy and dissipation energy were fixed as 10^{-4}. The modified date of all those files was used to mark as a benchmark for every simulation.

3.3. Run Calculation

Along with step size of one, the number of iterations was fixed as 1000. After that, the calculation was initialized and this process was continuously updated until the output was converged to get an optimal result. The different plots were obtained and various factors determined are inlet and outlet temperature, net wall flux, inlet and outlet pressure to determine the power required.

3.3.1. Analysis

In this work, there are three cases that have been analyzed such as 1 fin, 2fin and with 3 fin, to get the variation in the output temperature of the hot fluid. In this, we have analysed the heat exchanger with various numbers of fin and each with different angles to it. So that, we can disturb the incoming fluid and can vary the result.

4. RESULTS AND DISCUSSIONS

ANSYS software was used to conduct the numerical simulation for calculating the heat transfer and flow features of the present models. In this project, we are comparing the temperature differences between the three heat exchangers with difference in their internal fins. By that, we founded the results are as follows:

Case A: Geometry with 1 fin

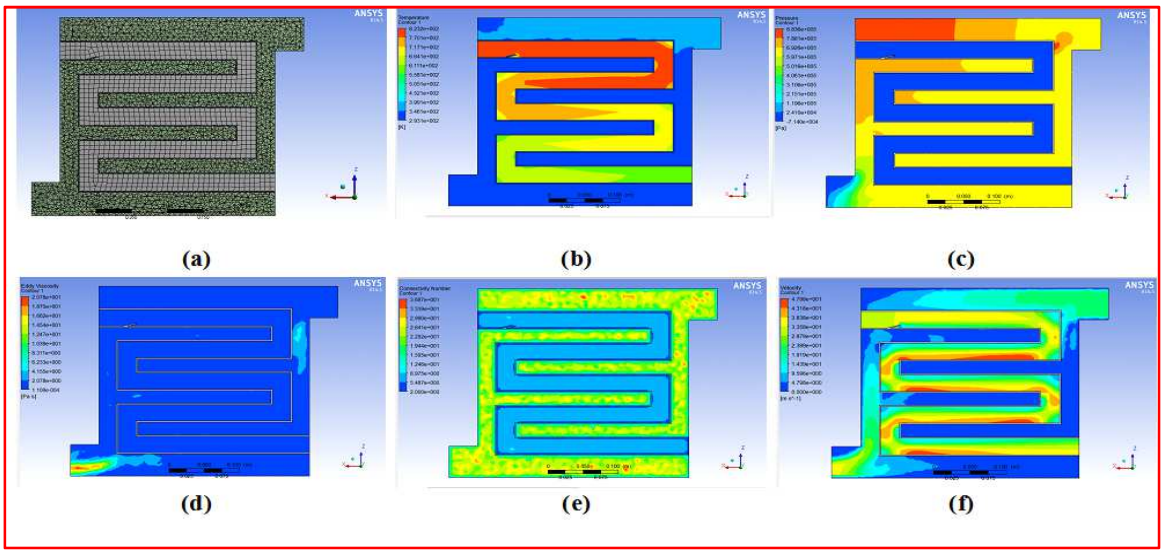


Figure 5: (a) Meshing the Geometry (b) Temperature (c) Pressure (d) Eddy Viscosity (e) Connectivity Number (f) Velocity.

Figures 5 to 7 provides the different contours such as temperature, pressure, eddy viscosity, connectivity number and velocity. The contour plot of pressure variation of CFHE was plotted in a vertical X-Z plane. The higher pressure zone has been observed at the nearer to the separation plate, where cold fluid is alienated from the hot fluid. Due to the major alteration in the cold fluid momentum, the higher pressure zone was observed at the upper side of the separation plate. At inlet, the cold fluid pressure inside the tube is maximum and reduced as it forwarded towards outlet of the tube. The higher pressure gradient was observed along the length of tube is mainly due to internal frictional contact between the fluid and tube. Due to the jet effect at the existing of cold fluid from the tube, a low negative pressure was attained. A gradual pressure gradient was observed at the hot stream of fluid. The pressure drop at the tube side was 250 Pa, which is higher than it was across tube bundle of 55 Pa.

The fluid flow plot of CFHE was shown in a vertical X-Z plane. The velocity of cold fluid was varied at the inlet of cold fluid in all the directions (X, Y and Z axis), which was produced by the frictional resistance of wall at the end of plate. The inlet fluid passes at high velocity which hits the separation plate, and then a part of cold fluid moves along the length of plate and remaining part of the fluid passes through the inside of the tube. The cold fluid velocity increases at the entrance of the tube because of the sudden contraction of large size of inlet into smaller size of tube.

The temperature distribution of CFHE was varied in a vertical X-Z plane. The plot of temperature distribution reported that the loss of heat of hot fluid was moved upward direction and heat is gained by the cold fluid. The cold fluid temperature was enhanced along its length of tube and observes heat from the hot fluid during the motion of fluid towards the outlet. The fluid coming from bottom surface of tube is hotter than the exit fluid, as the heat sources are placed at the bottom surface.

Case B: Geometry with 2fin

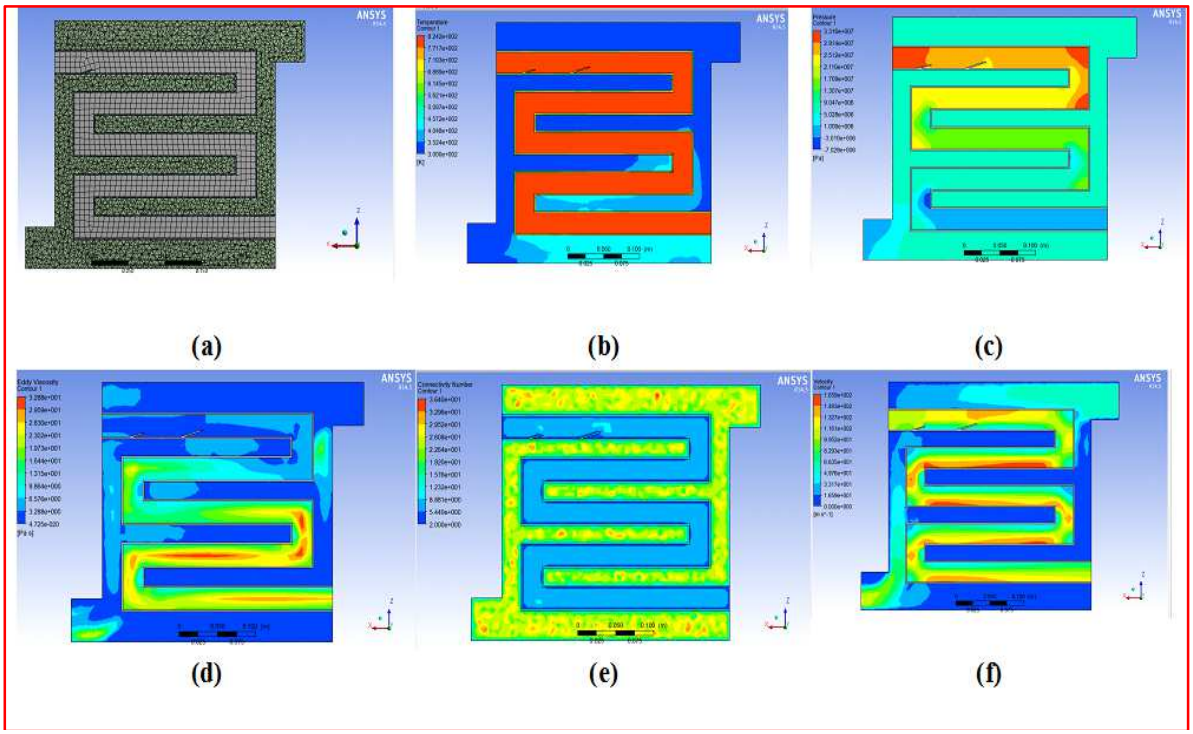


Figure 6: (a) Meshing the Geometry (b) Temperature (c) Pressure (d) Eddy Viscosity (e) Connectivity Number (f) Velocity.

Case C: Geometry with 3fin

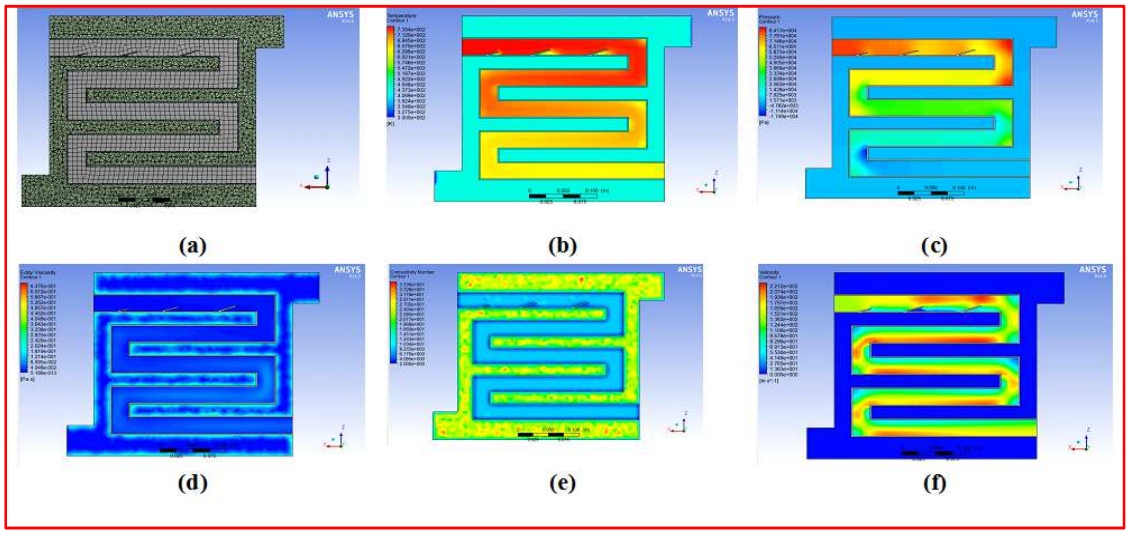


Figure 7: (a) Meshing the Geometry (b) Temperature (c) Pressure (d) Eddy Viscosity (e) Connectivity Number (f) Velocity.

Here, it is demonstrated that the coefficient of heat transfer coefficient is greater between the fin arrays. From these figures, it has resulted that there is a distribution of temperature gradient with a test tube and the difference in temperature difference are seem in all circumstances. Also, it was observed from the figures that the temperature gradient of the heat exchanger is greater than that of the smooth tube. This denotes that fining have a substantial outcome on enhancing the difference in temperature inside the test tube. The performance of heat transfer is created by the shape of fins based on the turbulence effect. The main important point is that, due to the higher conductivity of material for fins, which is increasing the rate of heat transfer. It has resulted that the fins are affecting this temperature distribution.

5. CONCLUSIONS

This research work aimed at computational measuring of the heat transfer behaviour in a cross flow heat exchanger for various fin configurations. The heat transfer reduces with the increasing of fin thickness, and it can also be analyzed by using numerical computation. The outputs are varied with the standard correlations for fins of constant thickness. The present work concentrates on the investigation of heat transfer and flow features for counter flow heat exchanger using ANSYS simulation software. The influences of the rate of mass flow in the inner tube as well as curvature ratio are examined in detail and the following conclusions are arrived: Nusselt number increases with enhancing of mass flow rate and curvature ratio value. In case of turbulent flow caused inside the pipe, the friction factor becomes reduced with enhancing of Reynolds number, but the rate of heat transfer improved with the Reynolds number. For increased rate of heat transfer, higher curvature ratio must be required regardless of power loss. From the temperature and velocity contour plots, it can be resulted that temperature increases towards inside of coil and velocity increase towards the outer side of the coil.

ACKNOWLEDGEMENTS

The authors gratefully acknowledge the support provided by the management of Saveetha Institute of Medical and Technical Sciences, Chennai, India for providing the necessary facilities to carry out this project. The authors also thankful to the necessary support, technical and valuable guidance provided by the supervisor C. Sathasivam, Assistant Professor, Saveetha Institute of Medical and Technical Sciences, Chennai.

REFERENCES

1. *A.N. Darvid, K.A. Smith, E.W. Merril, P.L.T Brain, Effect of secondary fluid motion on laminar flow heat transfer in helically coiled tubes, American Institute of Chemical Engineers, 17 (1971), pp.1142–1222.*
2. *J. Y. Jang, M. C. Wu., W. J. Chan, Numerical and experimental studies of three dimensional plate-fin and tube heat exchangers. International Journal of Heat and Mass Transfer 14 (1996), pp. 3057–3066.*
3. *Khudhair, M. R., & Mallarapu, M. G. K. Frequency Responses of Aluminum A356 Based on High Strength Alloy Composite (HSAP)*
4. *P. Wais, J. Taler, Fin shape optimization in tube heat exchangers by means of CFD program, 2nd International Conference on Engineering Optimization, Sept. 6–9, Lisbon, Portugal, (2010) pp. 1–10.*
5. *P. Wais, Fluid flow consideration in fin-tube heat exchanger optimization, Archives of Thermodynamics, 31, (2010) pp. 87–104.*
6. *W. M. Yan, P. J. Sheen, Heat transfer and friction characteristics of fin-and-tube heat exchangers, International Journal of Heat and Mass Transfer 43, (2000) pp. 1651–1659.*
7. *M. S. Mon, U. Gross, Numerical study of fin-spacing effects in annular-finned tube heat exchangers. International Journal of Heat and Mass Transfer 47, (2004) pp.1953–1964.*
8. *R. Romero-Mendez, M. Sen, K. T. Yang, R. McClain, Effect of fin spacing on convection in a plate fin and tube heat exchanger, International Journal of Heat and Mass Transfer 43, (2000) pp. 39–51.*
9. *P. Wais, Fin-tube heat exchanger performance for different louver angles, Zeszyty Naukowe Politechniki Rzeszowskiej Mechanika 86, (2014) pp. 115–122.*
10. *P. Wais, Influence of fin thickness and winglet orientation on mass and thermal efficiency of cross-flow heat exchanger, Applied Thermal Engineering 102, (2016) pp.184–195.*
11. *K.S. Bharuka, D.Y. Kasture, Flow through a helically coiled annulus, Applied Science Research, 41 (1984), pp. 55–67.*
12. *Nagaraja, R., Gurumurthy, B. R., & Shivanna, M. B. (2014). Bio softening of areca nut waste areca husk, leaf and leaf sheath for value added compost. Int. J. Res. Appl. Nat. Soc. Sci, 2, 105–112.*
13. *V. Kumar, S. Saini, M. Sharma, K.D.P. Nigam, Pressure drop and heat transfer study in tube-in-tube helical heat exchanger, Chemical Engineering Science, 61 (2006), pp. 4403–4416.*
14. *N. Ghorbani, H. Taherian, M. Gorji, H. Mirgolbabaei, Experimental study of mixed convection heat transfer in vertical helically coiled tube heat exchangers, Experimental Thermal and Fluid Science 34 (2010), pp. 900–905.*
15. *Bodade, P. R., Jogi, N., Gorde, M., Paropte, R., & Waghchore, R. Use of Internal Threads of Different Pitches to Enhance Heat Transfer in a Circular Channel.*
16. *R. Kharat, N. Bhardwaj, R.S. Jha, Development of heat transfer coefficient correlation for concentric helical coil heat exchanger, International Journal of Thermal Sciences, 48 (2009), pp. 2300–2308.*
17. *Qashqaei, A., & ASL, R. G. (2015). Numerical Modeling and Simulation of Copper Oxide Nanofluids Used In Compact Heat Exchangers. International Journal of Mechanical Engineering, 4 (2), 1, 8.*
18. *J.S. Jayakumar, S.M. Mahajani, J.C. Mandal, P.K. Vijayan, and RohidasBhoi, Experimental and CFD estimation of heat transfer in helically coiled heat exchangers , Chemical Engg Research and Design, 86 (2008), pp. 221–232.*

19. *P.B. Amar, H.M. Dange, Experimental studies of double pipe helical coil heat exchanger with micro fins, International Journal for Innovative Research in Science & Technology, 1 (2014), pp. 33–37.*
20. *T.J. Rennie, and V.G.S.Raghavan, Numerical studies of a doublepipe helical heat exchanger. Appl Thermal Eng, 26 (2006), pp. 1266–1273.*
21. *V. Kumar, B. Faizee, M. Mridha, K.D.P. Nigam, Numerical studies of a tube-in-tube helically coiled heat exchanger, Chemical Engineering and Processing, 47 (2007), 2287–2295.*
22. *K. Ravikumar, Ch.NagaRaju, MeeraSaheb, CFD Analysis of a Cross-flow Heat Exchanger with Different fin thickness, International Journal of Dynamics of Fluids, 13(2017), 345–362.*